Schematic Entry in OrCAD*

Patrick Frantz

This work is produced by The Connexions Project and licensed under the
Creative Commons Attribution License †

Abstract

A step-by-step tutorial to learn how to do a schematic entry in OrCAD.

WARNING: The Connexions version of the OrCAD tutorial is still in development. Please click here[1] for the original and complete tutorial. You may also browse this complete tutorial within Connexions by using the Mozilla browser and accessing the main Connexions page at http://cnx.rice.edu[2] . Click on the 'Contents' tab and select Rice University ELEC 424/427 under the 'Courses' tab.

1 Setting Up the Environment

Now that you have your parts library set up, you are ready to begin entering schematics. You can see from the project window that you already have one page of schematics called **PAGE1** in a folder called **SCHEMATIC1**. If you can't see this, click on the '+' next to the **dsn** file to expand the view. Even though our circuit is small enough to fit on one page, we will use two pages to demonstrate a schematic with multiple pages. First add a new schematic page by right-clicking the schematic root folder and selecting **New Page**. You will be prompted to provide a name for this page. Call it **Page 1 – Power and Connectors**. Now rename the original page by right-clicking and selecting **Rename**. Call this **Page 2 – Xilinx PLD**. It is always nice to give your schematic pages useful names. While you are at it, rename the schematic folder in the same manner, calling it **Elec424Tutorial**.

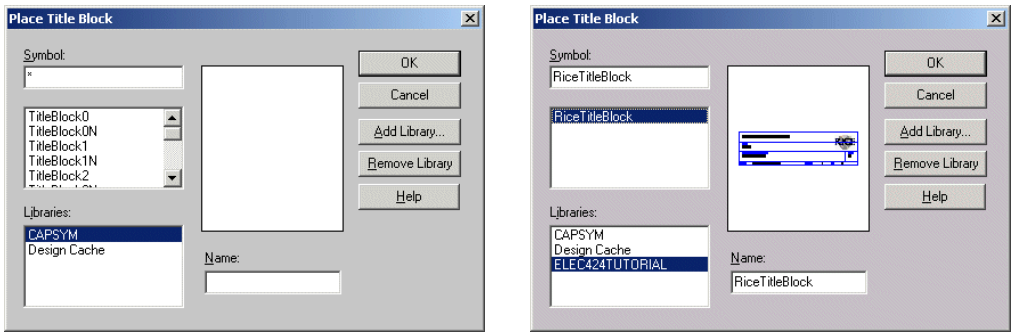
Open page 1 of your schematics by double-clicking it in the project window. This will bring up a blank page. Before we place parts, let's do a couple of things. First, I like to make the page size a bit bigger than the default. You can fit a lot more onto the page and it will still look nice when printed. To do this select **Options–>Schematic Page Properties**... Click the **Custom** radio button and use the following values (width=15.2, height=11.5). Second, there is a title block in the lower right corner. We are going to replace this with our own, so highlight the title block and delete it. We add a new title block by selecting **Place–>Title Block**... You will see the following dialog box.

*Version 1.5: Jan 29, 2004 2:34 pm -0600

†http://creativecommons.org/licenses/by/1.0

[1]http://koala.ece.rice.edu/index.cfm?page=pub

[2]http://cnx.rice.edu



(a) (b)

Figure 1

The title block is in your library, so we need to add that to the list of libraries. Click the **Add Library** button to browse to and select the library you created for the tutorial. Once you have added the library, you will be able to choose the **RiceTitleBlock** to add it to your design. Place the title block in the lower right corner of your schematic page. The title block has fields to put information for each page of schematics. Double-click the text to edit each field and change the information on each page so that it looks something like this.

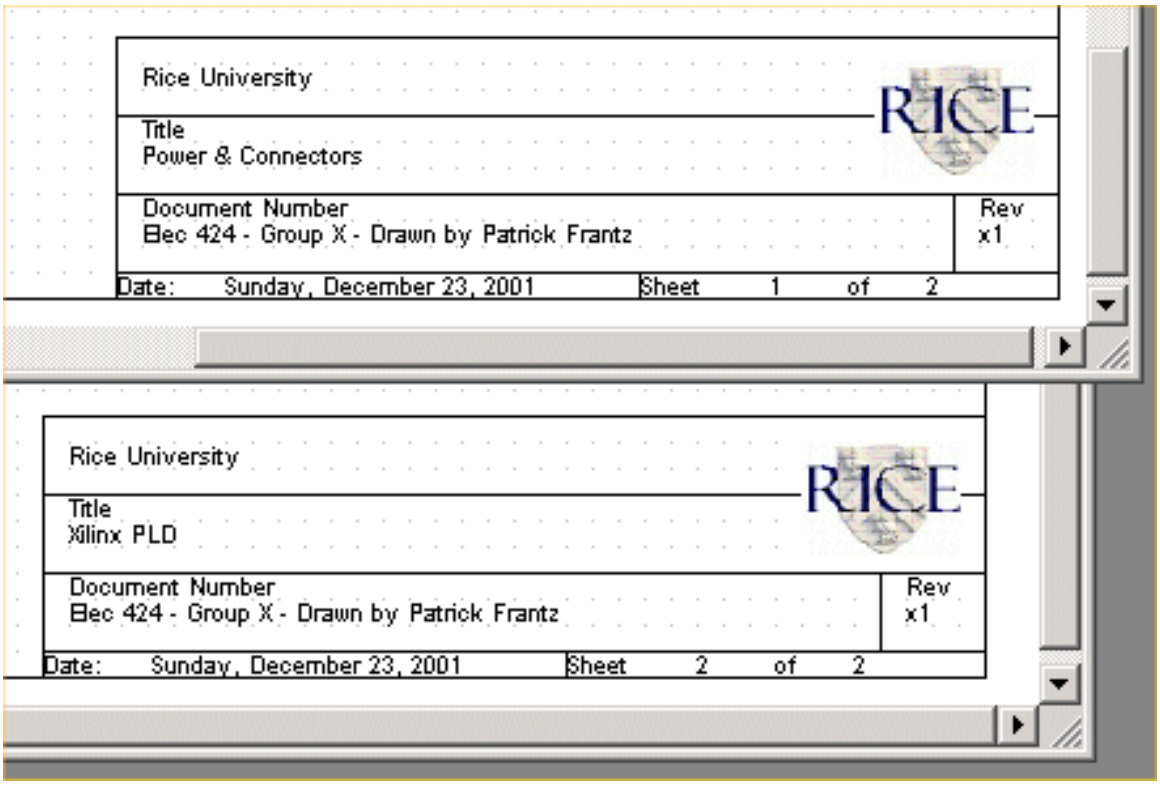


Figure 2

2 Placing Parts and Making Connections

You are now ready to start placing the electrical components for your design. Open the first page of your schematics and click the **Place Part** icon on the toolbar on the right side of the screen. You will then get a dialog for choosing which part you want to place on your schematics.

Figure 3: Place Part Icon

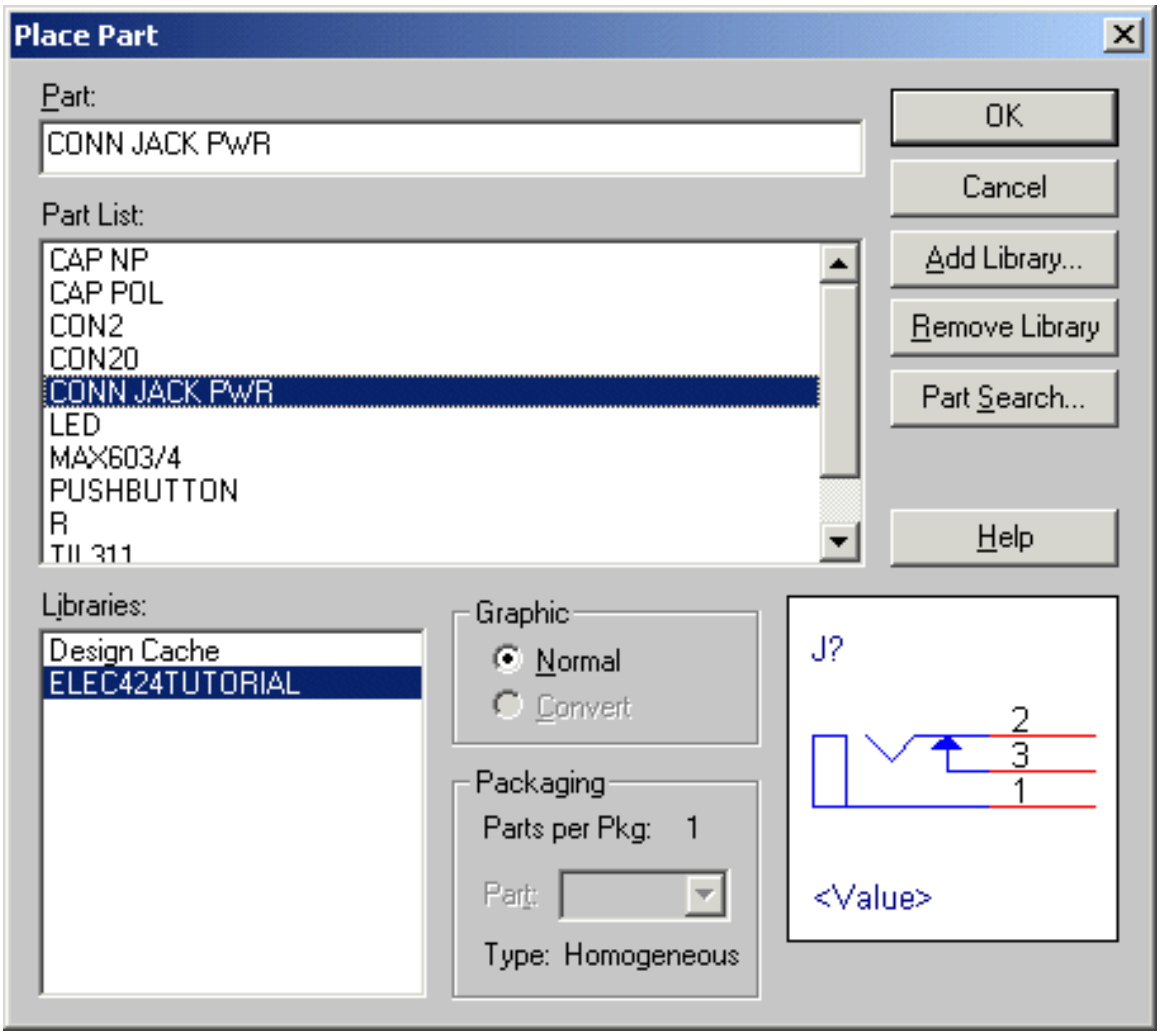


Figure 4: Place Part Dialog

Select the part **CONN JACK PWR** and click **OK**. Place the part on the left side of your schematic page. Now place the remaining parts on both pages using the attached completed schematics as a guide. A PDF file of the schematics can also be found on Owlnet.

`/home/jpfrantz/elec424/tutorial/sch/Elec424Tutorial.pdf`

A small hint for moving around in OrCAD: use '**I**' and '**O**' to zoom in and out, respectively. '**C**' will center the design at your cursor. '**R**' will rotate a part. You can do these actions while in the middle of another action (e.g. while placing a part). You will also notice that each part has a value associated with it. You can change this by double-clicking the current value. In this manner you can give all your capacitors, resistors, etc. the appropriate values. When you are done, the first page of your schematics should look something like this.

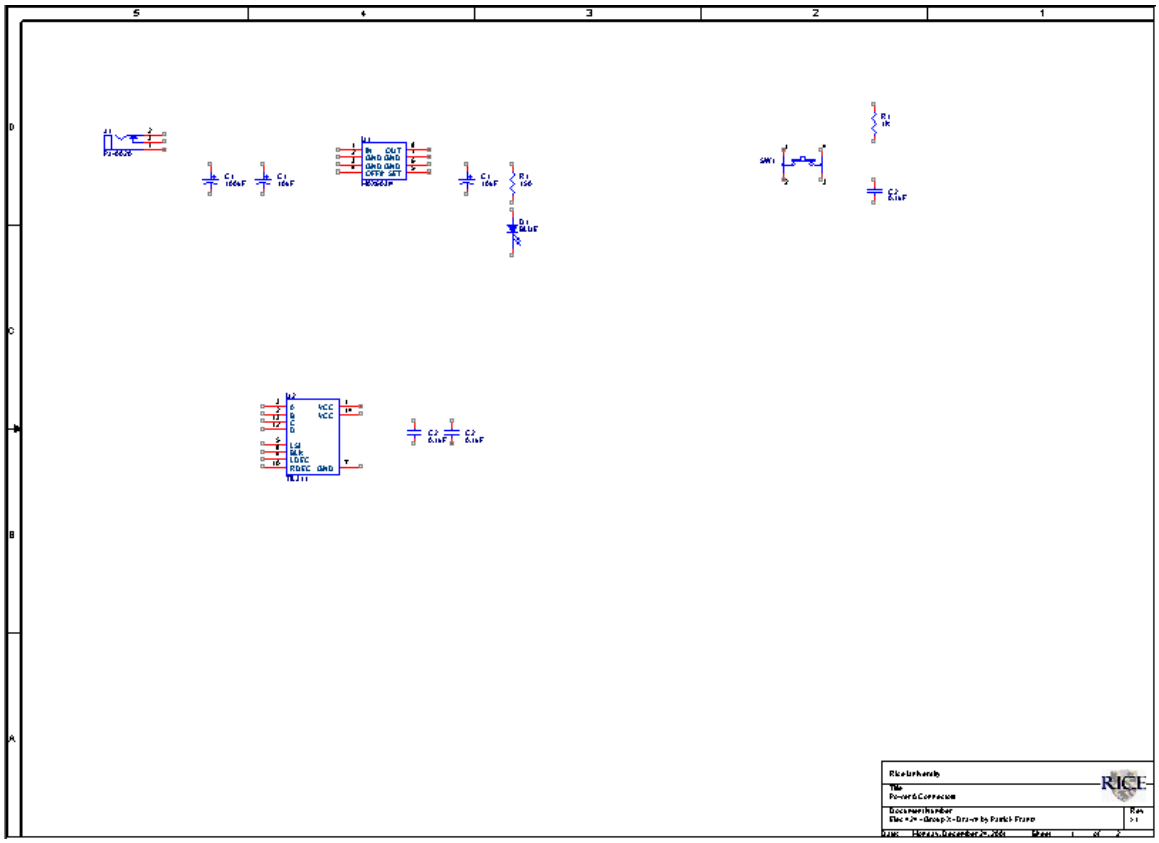


Figure 5

Now we need to draw nets to make electrical connections between components. To do this, click the **Place Wire** icon and connect the components as shown in the attached schematics. Use the **Place Bus** and **Place Bus Entry** icons to place busses and bus connections (you don't need to do this in these schematics).

(a) (b) (c)

Figure 6: (a) Place Wire Icon (b) Place Bus Icon (c) Place Bus Entry Icon

When you are done, the first page of your schematics should look like this.

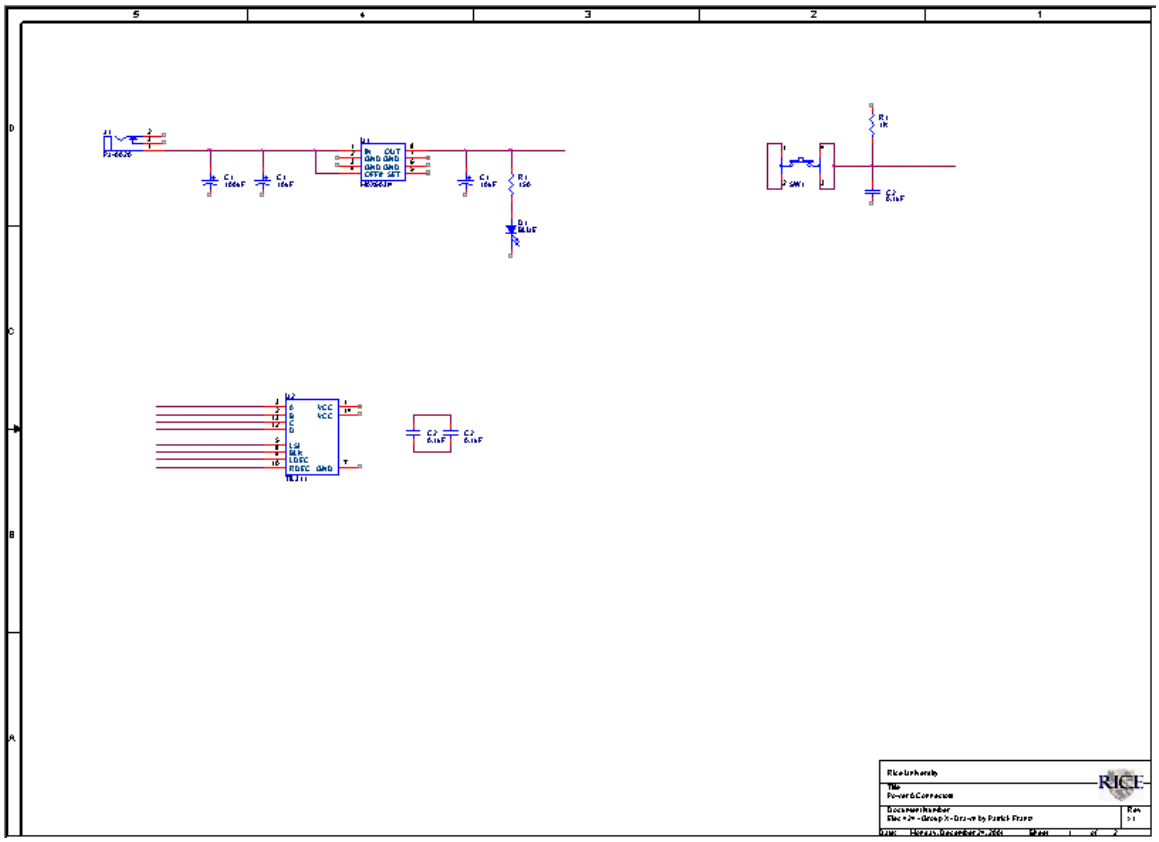


Figure 7

Now you need to add power and ground connections to some of the parts. OrCAD has several built-in symbols for power and ground. I like to use a symbol that explicitly names the nets, as shown in the picture below. I do this because many designs will have multiple power and ground nets. Explicitly naming them helps prevent shorts and other errors. It also makes your schematics easier to read. Add power and ground to your schematics now.

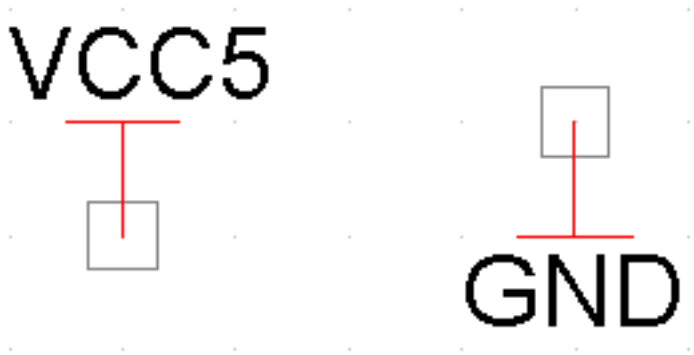


Figure 8

3 Connecting Pages and Naming Nets

Since some of these connections go to the PLD, we need to a way to connect the two schematic pages together. We can do this by using off-page connectors. To place these click the **Place Off-Page Connector** icon. Then select the connector called **OFFPAGELEFT-L** or **OFFPAGELEFT-R**, it doesn't really matter which one, they are functionally the same. You can place this on your schematic just like a part and then connect to it with a net. Off-page connectors are linked by a common name. For example, two off-page connectors on separate pages with the name CLK will be considered by OrCAD to be one net. To name a connector, just double click it to get a naming dialog box. Name your connectors now using the attached completed schematics as a guide.

Figure 9: Place Off-Page Connector Icon

NOTE: For reasons that will become clear later, I like to place my off page connectors as close to the edge of the page (right or left) as possible. This makes clear which nets go off page and which don't. It will also help you find mistakes on naming nets across pages.

Normal nets can also be named. This is extremely useful and can help tremendously in the layout process. I encourage you to use the **Place Net Alias** icon to name any important nets such as clocks, address and data bus lines, and other specific signals you are interested in. To name a net, first highlight the net you want to name and then click the icon, you will be given a dialog box to enter the name of the net. You can position the text anywhere you like. If you have already attached an off-page connector to a net, then that net already has the same name as the connector. You don't need to give the net an alias unless it will make your schematics more readable.

Figure 10: Place Net Alias Icon

If you have any pins on parts that are left unconnected, use the **Place No Connect** icon to mark it in your schematics.

Figure 11: Place No Connect Icon

Now complete your schematics as shown in the attached reference.